International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 4 Issue 6, September-October 2020 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

CFD Analysis for Heat Transfer Enhancement of a Corrugated Plate Heat Exchanger using Different Corrugated Channel Configurations

Prof. Pushparaj Singh[1], Rishi Kesh Jha[2]

[1]Assistant Professor, [2]M Tech Scholar,
[1,2]Rewa Institute of Technology, Rewa, Madhya Pradesh, India

ABSTRACT
The flow properties of heat transfer across such corrugated channels are very distinct from parallel plate channels. The primary flow direction is parallel to the waviness of the channel of the corrugated channel, although the local flow direction is still changed due to the waviness of the channel. By flow recirculation, separation and re-attachment, the thermal boundary layer formed on its walls is regularly disrupted and the convective heat transfer coefficient is thus increased. In this work, in corrugated channels, the convective heat transfer and pressure drop characteristics of flow have been tested numerically. On channels of uniform heat flux and fixed corrugation over a Reynolds number spectrum, 10000 ≤ Re ≤ 30000, numerical research was conducted. The impacts of channel configuration, variation in Reynold's number, and concentration variations of nanoparticles in base fluid on heat transfer are discussed. As opposed to the traditional corrugated channel, a major heat transfer gain was associated with the proposed corrugated channel. Based on the nanofluid concentration, Reynold number and channel configuration of the corrugated channel, the average Nusselt number increased by a factor of 2.2 up to 2.8 compared to that of the semi-circular corrugated channel. Compared with the effect of channel configuration variation, triangular corrugated channel was the best Nusselt numbers, followed by the shape (Semi-circular + Triangular), and Semi-circular corrugated channel with the lowest results.

KEYWORDS: *Nanofluids, Corrugated channels, Reynold's number, Heat transfer, Nusselt Number, CFD*

***How to cite this paper**:* Prof. Pushparaj Singh | Rishi Kesh Jha "CFD Analysis for Heat Transfer Enhancement of a Corrugated Plate Heat Exchanger using Different Corrugated Channel Configurations" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-4 | Issue-6, October 2020, pp.1524-1532, URL: www.ijtsrd.com/papers/ijtsrd35706.pdf

IJTSRD35706

I. INTRODUCTION

To design better energy systems, researchers and engineers would minimize energy usage to increase the performance of the energy system. The improvement of heat transfer rates and thus output, which is important components for many technological applications such as space, aeronautics, car industry, ocean thermal energy transformation technology is a major concern in this regard. In this sense, the improvement of heat transfer rate is important. In the heat exchangers, corrugations are used to increase the transfer rate and the strength of plates. The complex geometry of the channel corrugated improves the efficiency of heat transfer, contributing to increased pressure losses, especially in turbulent flow.

Flow control methods include three primary methods: active flow control, passive flow control and compound flow control for heat transfer rate improvements. The ***active flow control technology*** requires external power input to increase the heat transfer. For e.g., flow oscillation, flow vibration, surface vibration, magnetic field and other related techniques are active flow methods. This example offers better mixing of flow and increased heat transfer. In the ***passive flow control method***, no external input is required to enhance the heat transfer; but, due to geometric adjustments, there is a further pressure decrease. The use of inserts, additives, rough surface, swirl flux systems as well as treatment surfaces and extended surface areas as coiled tubes are some examples of passive flow control approaches. The reduction of the hydraulic diameter of the flow passage increases the transfer rate, as can be seen from examples. In some cases a secondary flow can be achieved using this method, which increases the heat transfer rate through the mixing of fluids between the central flow area and the flow area near the wall surface. The technique of ***compound flow control*** requires combinations of two or more systems of flow control for heat transfer. An example of the compound flow control technique might be a surface structure with additives or flow vibration with additives.

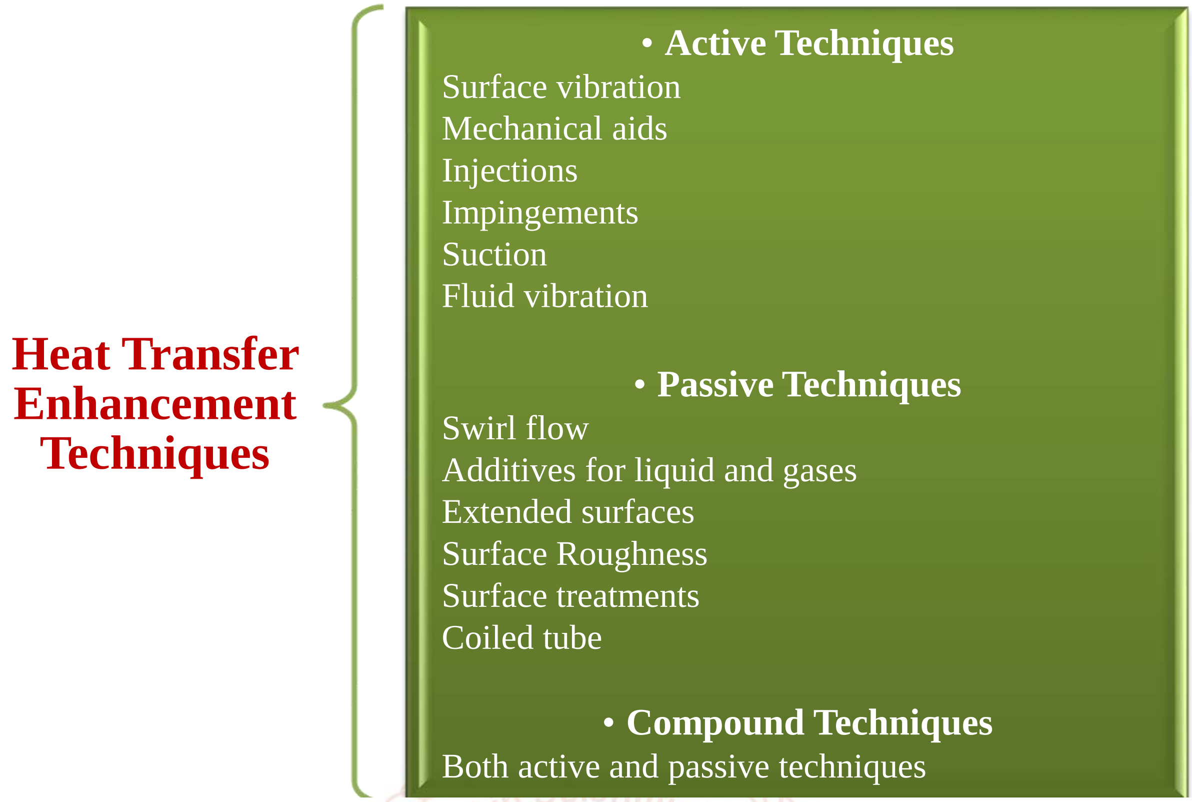

Figure 1 Enhancement techniques for heat transfer.

The corrugated channels are significantly used in industrial heat exchangers as a passive flow control technology to increase the heat transfer rate. This technique is designed to interrupt and reform the flow boundary layer through the channel. Present work presents an investigation on the heat transfer enhancement of a corrugated channel using different corrugated configurations in CFD. The reasons for this analysis are that very few publications on the heat transfer enhancement using different shaped corrugation channel are published.

II. Literature Review

During several research projects in recent years, the heat transfer performance of the corrugated channel was the object of computational, theoretical and experimental work. Most work has been performed to increase the resulting heat transfer coefficient in order to improve the heat transfer of corrugated plate heat exchanger. Several studies on corrugated channels have been performed and their thermal and hydraulic characterization data are available in open literature. In these recorded similarities, however, there is a widespread difference and it was appropriate to examine the experimental facilities and procedures, methods of data reduction, findings and assumptions of some of the relevant past works before starting the present research.

2.1. Previous work

Mohammed and Abed (2008) in a corrugated channel, laminar forced convection heat transfer and fluid flow characteristics have been numerically tested. The channel walls' temperature was kept stable, which was greater than the temperature of the fluid. The effect of the wavy angle and number of Reynolds on fluid flow and heat transfer has been studied. For the solution, the range of the Reynold number was found to be 500 to 2500, wavy angles ranged from 0 ° to 60 ° and the number of Prandtl was 0.7. The maximal values of heat transfer enhancement and pressure decrease were observed to be 3.6 and 1.11 times greater than those at wave angle λ = 40 ° from the plane channel, respectively. [1].

Khan and Kumar (2009) described performance and exergy of corrugated plate heat exchanger in parallel or in counter flow. Plate had sinusoidal wavy surface with corrugation angle of 45°C. Heat exchanger contained 3 Channels. Hot fluid flow at the middle channel which was cooled by water through outer channels. Hot water temperature was in the range of 40°C to 60°C. Reynolds number was in the range of 900< Re>1300 for hot and cold fluid. After performing experiment performance or effectiveness of corrugated plate heat exchanger for counter flow was found out to be 44.5% more as compared to parallel flow arrangement. As well as exergy loss in counter flow is 7.2% less as compared to parallel flow [3].

Kumar et.al (2010) has made an attempt to investigate the performance and effectiveness of corrugated plate heat exchanger. Experiment was conducted on three channels 1-1 passes of corrugated plate heat exchanger. Hot fluid was made to flow at the middle channel while the cold fluid flow at top and bottom channel in counter and in parallel flow. Plate had a sinusoidal shape at an angle of 30° corrugation angle. Temperature of hot fluid was in the range of 50°C to 70°C whereas temperature of the cold fluid was in the range if 30°C to 40°C inlet. It was found that the effectiveness of counter flow heat exchanger is 48% higher than the parallel flow. As well as exergy loss was also calculated and found 33% less in counter flow arrangement as compared to the parallel flow arrangement [5].

Faizal and Ahmed (2012) performed an experiment on corrugated plate heat exchanger for small temperature difference applications. It had 20 corrugated plates placed parallel to each other and its total heat transfer area was 1.16298 m^2. The spacing between the plates varied 6 mm, 9 mm, and 12 mm. Minimum rate of heat transfer was obtain when distance between plates was 6 mm. Water had been used as a hot and cold fluid which flow in alternate channels. In this experiment effect on rate of heat transfer was determined while varying flow of hot water whereas the cold side flow rate and the hot and cold water inlet temperatures were kept constant. In the result it was found that when the mass flow rate of hot fluid increased corrugated plate enhance turbulent at higher velocity which increases heat transfer rate. The overall heat transfer coefficient U, the pressure losses and the average thermal length are found to increase with increasing hot fluid flow rate and heat transfer rate of heat exchanger with 6 mm heat exchanger compared to other values. The plate heat exchanger with 6 mm is found to be appropriate due to effective high thermal length and heat transfer when the pressure loss is higher [8].

Dnyaneshwar et.al (2013) focused on the modeling a copper plate heat exchanger for milk pasteurization in a food industry using high temperature for a short time. This paper presents analytical and CFD analysis of pressure drop of counter flow for milk and water over copper and steel plate type heat exchanger for determining the energy required for circulating fluid. Knowing all operations parameters problem was first solved theoretically by LMTD. After that comparison between CFD and analytical result it was done. It was found that energy required for the circulation of water & milk is very low in copper plate type of heat exchanger as compared to the steel plate type of heat exchanger [10].

Giurgiua et.al (2014) numerically studied two different models on plate heat exchanger. Geometry of plate influence rate of heat transfer. One plate heat exchanger contains mini channels at 30° while other plate heat exchanger contains mini channels at 60°. In result from CFD and numerical analysis it has been found that plate heat exchanger with 60° mini channels give high rate of heat transfer as compared to 30° mini channel heat exchanger [11].

Kumar et.al (2014) investigated the performance of baffle shell and tube heat exchanger by using CFD tool ANSYS. The work was carried out to determine the performance of heat exchanger by changing the inclination of baffles in shell and tube heat exchanger. Three different baffles inclination angles namely 0°, 45° and -45° were used in CFD modeling to find the impact of baffle inclination angle on the characteristics of heat transfer and also on fluid flow. As the result of CFD comes out it had been observed that the steady state heat flux comes out to be more in the case of +45 degree baffles case than -45 degrees baffles case. The heat flux comes of 0 degree baffles come out intermediate between 45 and -45 degrees cases [12].

Hasanpour et al. (2016) have experimentally studied a double pipe heat exchanger with inner tube corrugated filled with various categories of twisted tapes from conventional to modified types (perforated, V-cut and U-cut). The twist ratio, the hole diameter, the width and depth ratio of the cuts have been varied and the Reynolds number has been changed from 5000 to 15000. Overall more than 350 experiments were carried out. Nusselt number and friction factor for corrugated tube equipped with modified twist tapes are found out to be higher than typical tapes [15].

Johnson et.al (2017) studied the analytical design of the heat exchanger which has been also numerically analyzed. On the basis of standard k- ε modelling CFD analysis have been done. The solution of the problem yields when the optimum values of flow rate, outer diameter of pipe and inner diameter of pipe to be used at an effective length for a double pipe heat exchanger. When the stream processes for specified flow rates then it was treated for a given inlet to outlet temperature. From the result it has been found that the design and analysis of the double pipe heat exchanger would be a great success [16].

R K Ajeel et.al (2017) studied CFD study on turbulent forced convection flow of Al_2O_3-water nanofluid in semi-circular corrugated channel. Computational Fluid Dynamics (CFD) simulations of heat transfer and friction factor analysis in a turbulent flow regime in semi-circle corrugated channels with Al_2O_3-water nanofluid is presented. Simulations are carried out at Reynolds number range of 10000-30000, with nanoparticle volume fractions 0-6% and constant heat flux condition. The results for corrugated channels are examined and compared to those for straight channels. Results show that the Nusselt number increased with the increase of nanoparticle volume fraction and Reynolds number. The Nusselt number was found to increase as the nanoparticle diameter decreased. Maximum Nusselt number enhancement ratio 2.07 at Reynolds number 30,000 and volume fraction 6% [17].

Junqi et al. (2018) has experimentally investigated the thermal hydraulic characteristics for three types of fluids (R245fa, glycol & water) on plate heat exchanger surface. To overall evaluate the enhanced heat transfer, concept of pump power is provided. Using multiple regression method, dimensionless correlation equation of Nusselt number & friction factor are given. It is concluded that the plate chevron angle affect thermal hydraulic performance. Heat transfer increases with increase in chevron angle & vice versa [18].

2.2. Problem finding of review

The heat transfer is one of the most important criteria for measurement as thermal potential and efficiency for heat transfer of the heat exchange plate can be seen from a study of various literature reviews. The highest thermal transfer efficiency has been found among all types of heat exchangers. Corrugated channels in several thermal channels, including the internal cooling channels used in gas turbine blades in the past years, have been a common amplification of heat transfer. These technologies demonstrate that there has been substantial change in the flux mixing between hotter flow layers next to the channel wall and cooler flow layers in the middle region. This leads to a significant decrease in the flow of turbulent energy from the corrugations and hence an increase in turbulent heat transport in the channel. Corrugated heat exchangers are used for heat transfer; however, their construction is very typical in comparison to heat exchangers. Furthermore, fluid research computer and experimental analyses have been used to execute a number of experiments. Many studies have studied various aspects of the corrugated channel system to improve heat transfer. However, many other corrugated structures also have to report their performance in heat transfer. Moreover, very few experiments investigated the effects on the corrugated channels of geometrical parameters.

III. Research objectives and Methodology

In this study, a 3-dimensional numerical (3-D) simulation was used to investigate the thermal characteristics of the corrugated plate heat exchanger, with different corrugated channel configurations. ANSYS 17.0 simulation software was used to research the physiognomy of the heat transfer physiognomies of a corrugated plate heat exchanger, with different corrugated channel configurations.

The main objectives of the present work are as follows:

- To examine the thermal characteristics of a different corrugated channel configurations of the corrugated plate heat exchanger.
- In order to build and test the corrugated channel model on a CFD model, comparisons will be made with the previous model.
- Thermal characteristics such as Nusselt number, and heat transfer coefficient are evaluated in the effect of a different corrugated channel configurations.
- Calculation of flow rate variance effects for the output of a different corrugated channel configurations of the corrugated plate heat exchanger.

3.1. Computational Fluid Dynamics

The computer-based research is computational fluid dynamics. It helps one to analyse different aspects in chemical reactions, such as liquid flux, pressure distribution, heat transfer and the phenomena. For the processing of CFD simulations, there are three main elements: pre-processor, solver, and postprocessor.

Pre-processor: A pre-processor is defined to the geometry regarding the problem. And it is fixed into the domain for the computational analysis and then yields the mesh associated with geometry. Here also put the nomenclature like inlet, outlet, and wall etc. Usually, the finer the mesh associated with geometry into the CFD analysis offers more solution that is accurate. Fineness for the grid additionally determines the computer hardware and much more time needed for the calculations.

Solver: - The calculations is done by using the numerical solution methods in the solver processor. You will find the countless numerical practices that are utilized for the computations for example:-the finite factor method, finite amount technique, the finite huge difference technique additionally the spectral strategy. Most of them in CFD codes use finite volume method. The FVM is used in this project. The solver perform the steps: that is following

- Firstly the fluid movement equations are integrated on the control volumes (leading to the actual preservation of appropriate properties for every finite amount),
- Then these key equations tend to be discretised (creating algebraic equations through converting of the fundamental fluid movement equations),
- And then finally an iterative method is used to fix the algebraic equations.
- Pressure based paired option method CFD rule is used for re solving the simulations in this task.

Post-Processor: The post-processor is offered to the visualization for the total link between the solutions. It includes the ability to display the mesh and geometry also. As well as in this processor we could create the vectors, contours, and 2D and surface that is 3D of the issue solutions. Right Here the model can also be manipulated. In this method we could also look at cartoon of the problem.

3.2. Steps taken during the analysis

This mentions the steps that have been taken place to achieve the objectives of the work.

- We first developed the CFD model of different corrugated channel on ANSYS 17.0 for CFD research.
- Meshing of model is done on CFD pre-processor.
- The boundary conditions are applied on the model and numerical solutions are calculated by using solver.
- The FVM is used in solving the problem.
- The solution is calculated by giving iterations to the mathematical and energy equations applied on model.
- Validation will be carried on CFD model with previous model.
- Applying formulas for calculating Nusselt number and friction factor.
- The results can be visualized in the form contours and graphs by CFD post processor.
- Result analysis.

3.3. Calculation procedure

The Reynold's Number (Re) can be expressed as:

$$Re = \frac{\rho \times u_{in} \times D_h}{\mu}$$

The velocity of the inlet was given by:

$$u_{in} = \frac{Re \times \mu}{\rho \times D_h}$$

The heat received by the nanofluids from the test section can be determined as follows:

$$q'' = \dot{m} c_p \,(T_{in} - T_{out})/A$$

The average heat transfer rate gives the average heat transfer co-efficient which is given by:

$$h = q'' \frac{ln\left(\frac{T_w - T_{in}}{T_w - T_{out}}\right)}{(T_w - T_{in}) - (T_w - T_{out})}$$

The Nusselt number can be calculated from the following expression:

$$Nu = \frac{h\, D_h}{K_{nf}}$$

IV. Geometry Setup and Modelling

The study uses the CFD model in this section to examine the heat transfer physiognomies of the corrugated channel with a different configurations. CFD analysis involves three major steps: (a) pre-processing, (b) solver execution, and (c) post-processing. The first step includes the creation of the geometry and mesh generation of the desired model, while the results are seen as expected in the last step. In the execution of the solver (medium) stage, the boundary conditions are fed into the model.

4.1. Geometry Setup

The geometry of the corrugated channel conducted in the simulation study is drawn from **R K Ajeel et.al (2017)[17],** a research scholar with exact sizes. After the planned prototypes, the different configurations in the shape of triangular, and mixed (triangular+semi-circular) corrugation is applied to the platform of corrugated channel. Figure 1. Demonstrates the geometry of the corrugated channel. The numerical model for different corrugated channels is seen here as the test section and in Figure 2, 3, or 4. The overall length of the channel is L_{total}=700 mm. The test section is long L_2=200 mm and the upstream rectangular part L_1=400 mm, upstream indicates the flow has been extensively formed at the leading edge. In the downstream segment, the L_3=100 mm width is used to prevent adverse pressures caused by reverse flow from occurring in trailing areas. The channel's height (H) is 10 mm and the channel's width (W) 50 mm. The geometry architecture was carried out with designer Ansys 17.0.

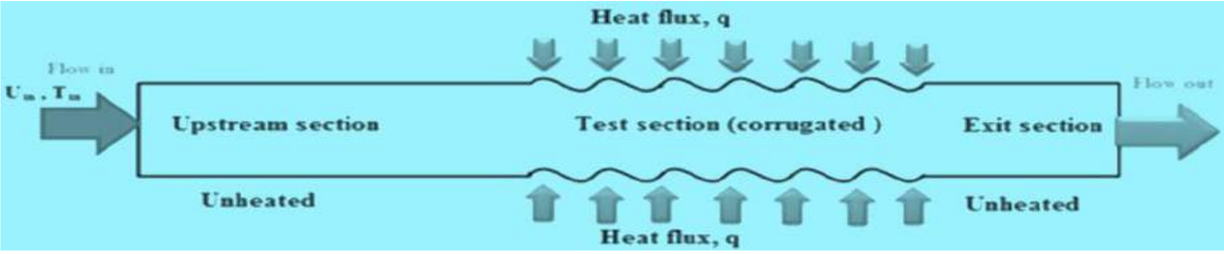

Figure 2 The current study's physical model

The semi-circular corrugated channel width (w) =5 mm with fixed pitch (p= 15mm).

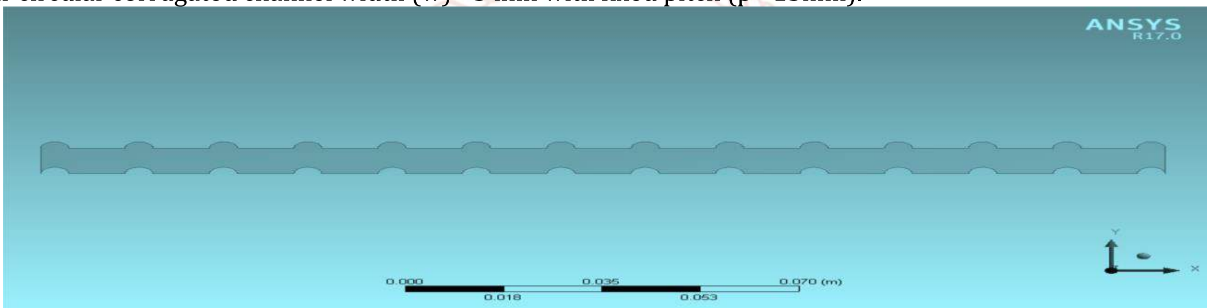

Figure 2 The geometrical model of Semi-circular corrugated channel (test section) conventional design

Triangular corrugated channel width (w) = 4.75 mm at fixed pitch (p = 15 mm).

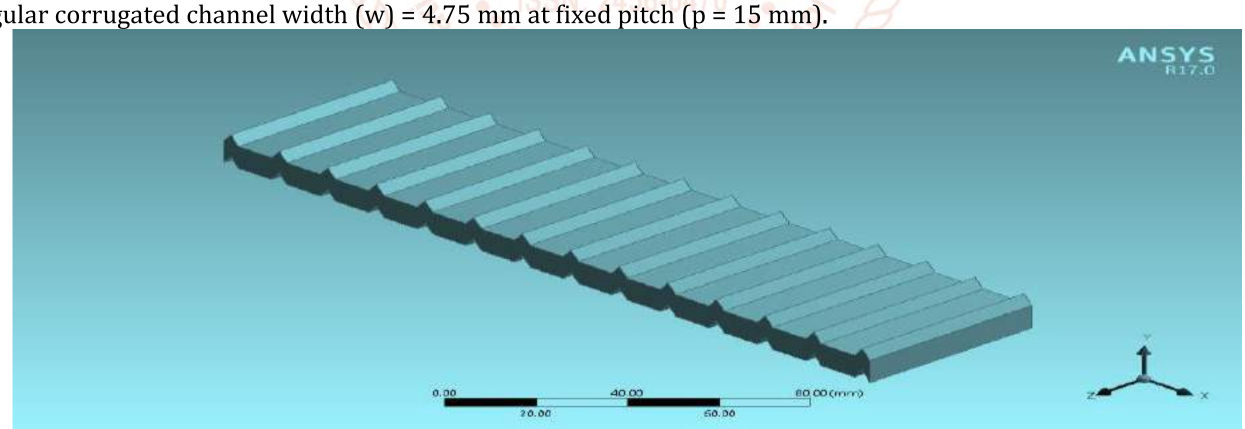

Figure 3 The geometrical model of triangular corrugated channel (test section) proposed design

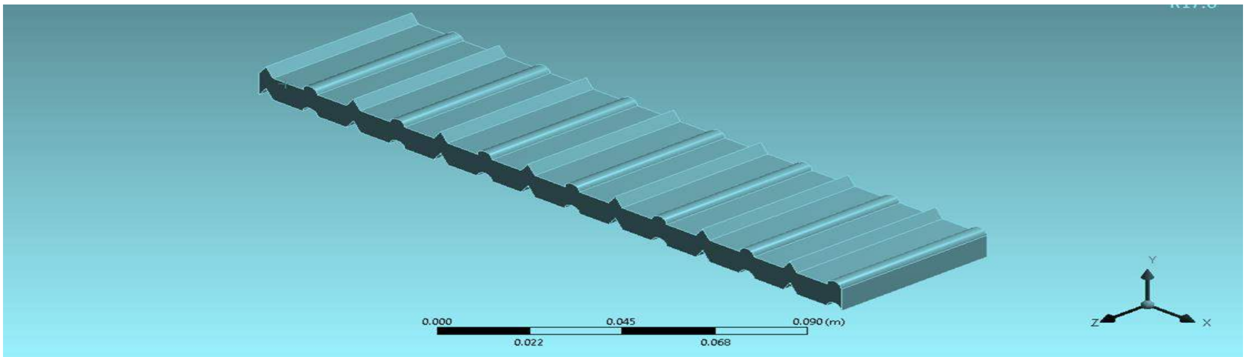

Figure 4 The geometrical model of Semi-circular+Triangular corrugated channel (test section) proposed design

4.2. Meshing
Mesh generation was performed by using Ansys Meshing modular. In the ANSYS FLUENT R17.0 pre-processor stage, a three-dimensional discrete corrugated channel model with different configurations was developed. While grid types are related to simulation output, the entire system is divided into the finite volume of Quadcore tetrahedral grids in order to reliably measure the thermal physiognomies of the corrugated channel model with different configurations using the correct grids.

Table 1 Meshing Details

MODEL TYPE	NODES	ELEMENTS	MESH TYPE
Semi-circular	132804	118400	Tetrahedral & quad core
Triangular	118250	102459	Tetrahedral & quard core
Semi-circular + Triangular	135256	118716	Tetrahedral & quad core

4.3. Model Selection and Solution Methods
Fluent 17.0 has been used to calculate computationally. In science, the method used to separate the governing equations was a finite element. For this convective word, the scientists have used a simpler algorithm, for the coupling with the pressure velocity, the SIMPLE algorithm. The second upwind mechanism has been introduced and the normal wall element is used in the conventional turbulent configuration of k-s. The convective thermal transfer characteristics can be performed by solving directed equations i.e. Navier Stokes for energy, mass and constant thermal flow respectively continuity and energy measurements.

4.4. Material Property
Table 2 Thermophysical properties of distilled water and Al_2O_3–water nanofluid with different volume fractions at 300 K

$\varnothing(\%)$	$\rho(\frac{Kg}{m^3})$	$c_p(\frac{J}{Kg-K})$	$K(\frac{W}{m-K})$	$\mu(\frac{N-s}{m^2})$
0	998.2	4182	0.6	0.001
2	1050.236	3947.74	0.6355	0.001273
4	1102.272	3735.60	0.6718	0.001650
6	1154.308	3542.59	0.7083	0.002139

4.5. Boundary Conditions
Machine domains and boundary conditions were introduced in the various corrugated channels, including velocity input and temperature of 300 K, pressure outlet and the slide speed. In actual fact, standard heat flux (q= 10 KW / m^2) was applied to the corrugated walls, while most of the straight walls were adiabatic. Below you can see simple thermal conditions and limits of the dynamic flow area. The fluid velocity at the inlet can be determined for different values, according to the number of Reynolds. The flow is represented by five Reynolds numbers, 10000, 15000, 2000 and 30000. For the testing field, the top and bottom walls are heat streaming continuously of 10 KW / m^2, while the residual walls are adiabatic. The inlet temperature of the working fluid is T_{in}=300K. The Al_2O_3/water nano-fluid flows continuously over the test region of the Reynolds numbers mentioned above.

V. Results and discussions
This section is designed for measuring the thermal performance of the different configurations of corrugated channel. Nusselt and heat transfer coefficient variations are measured at different Reynold's number and different concentration of nanofluids for the investigation of corrugated plate heat exchanger.

5.1. Validation of numerical computations
For validation, In order to numerically validate the CFD model of heat exchanger for initial case we considered water as a working fluid. Here water flows as different Reynolds number inside the semi-circular corrugated heat exchanger and the results obtained by simulation for semi-corrugated channel which included Nusselt number (Nu) are compared with **R K Ajeel et.al (2017)** results [17]:

Table 3 shows the values of Nusselt number calculated from the CFD modeling compared with the values obtained from the analysis performed by R K Ajeel et.al (2017) [17] for semicircle corrugated channel using water as a working fluid

S. No.	Reynold's Number	Nusselt Number (Base Paper)	Nusselt Number (Present Study)
1.	10000	124	124.12
2.	15000	183	183.18
3.	20000	239	241.41
4.	25000	288	290.50
5.	30000	348	348.08

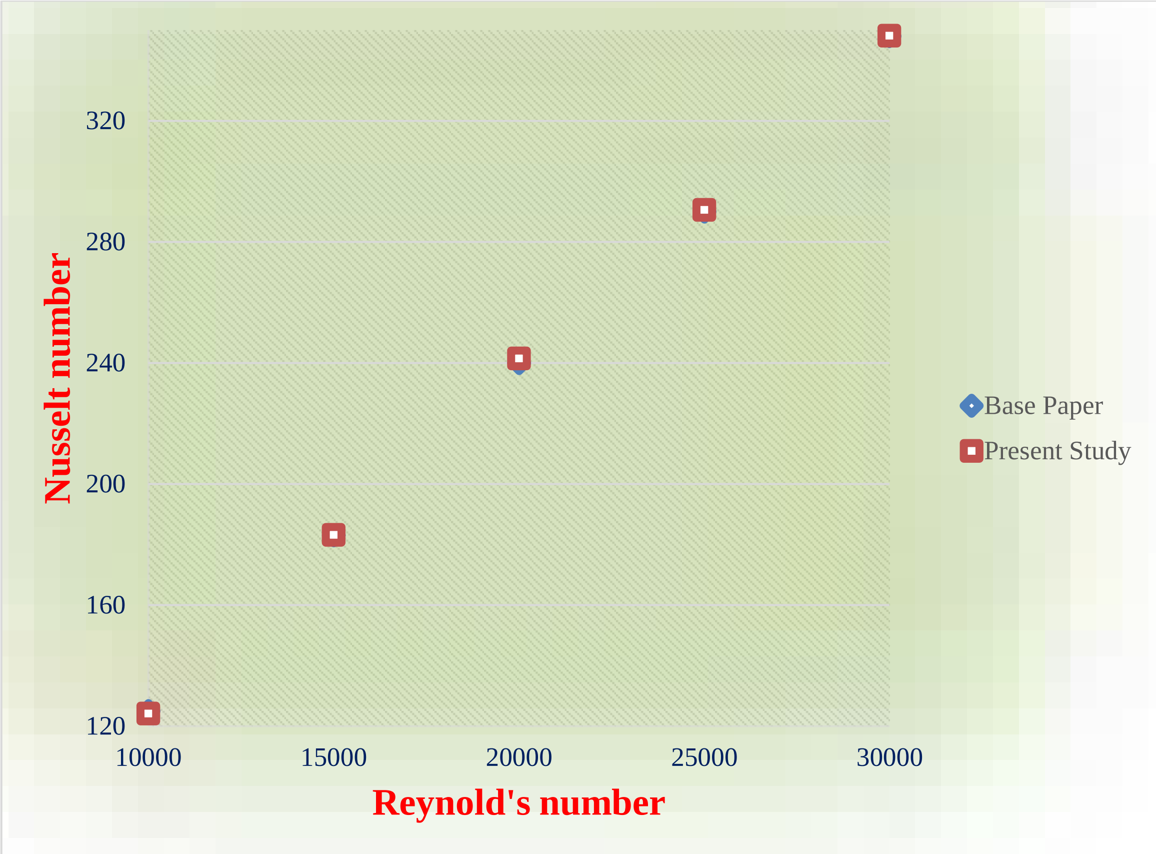

Figure 5 Values of Nusselt number calculated from the CFD modeling compared with the values obtained from the analysis performed by R K Ajeel et.al (2017)

From the above analysis it is found that the value of Nusselt number calculated from CFD analysis is close to the value of Nusselt number obtained from the base paper. So here we can say that the CFD model of corrugated channel is correct.

5.2. The effect of different types of corrugated channels on the flow and thermal characteristics in the corrugated heat exchanger

The turbulent forced convection of Al_2O_3-water nanofluid in different corrugated channel were investigated. The CFD results obtained due to the impact of the different models, ϕ and Reynold's number have been displayed in terms of Nusselt number.

5.2.1. Effect of Triangular corrugated configuration on the flow and thermal characteristics in the corrugated heat exchanger

In this the 3-D turbulent forced convective flow of Al_2O_3 –water nanofluid for triangular corrugated channels over Re ranging from $10{,}000 \leq Re \leq 30{,}000$ were examined under constant heat flux while ϕ ranged from 0 to 0.06

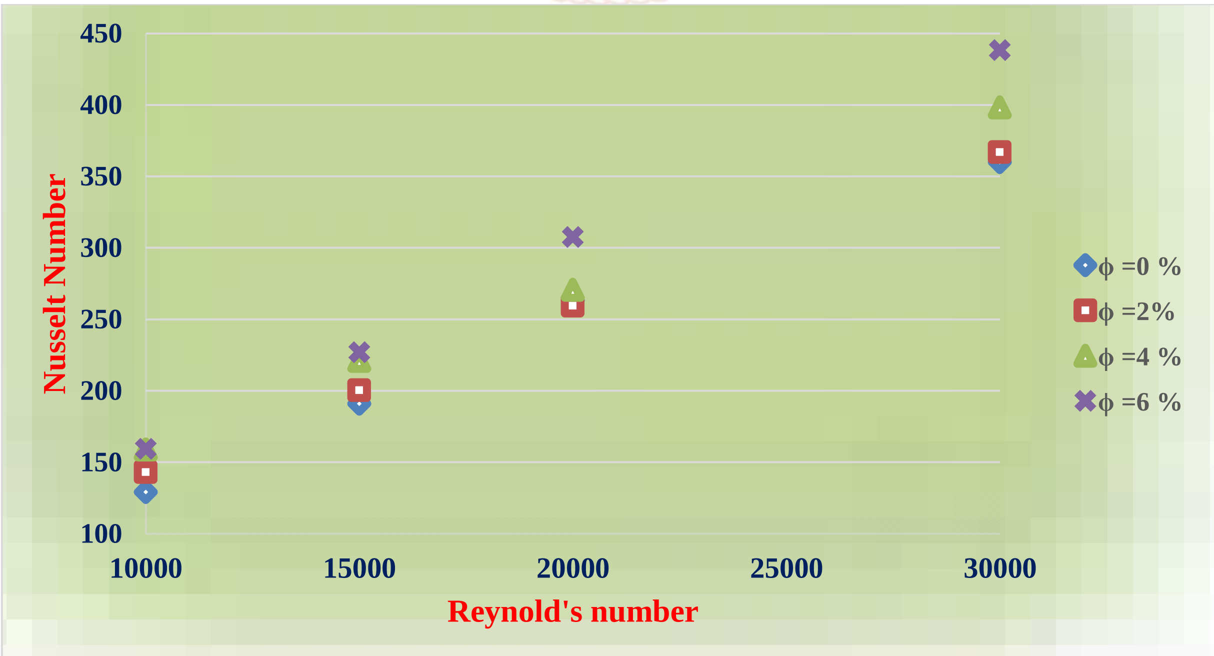

Figure 6 Value of Nusselt number calculated through CFD analysis at different Reynold's number for Triangular corrugated configuration under constant heat flux while ϕ ranged from 0 to 0.06

5.2.2. Effect of Semi-circular + Triangular corrugated configuration on the flow and thermal characteristics in the corrugated heat exchanger

In this the 3-D turbulent forced convective flow of Al_2O_3 –water nanofluid for mixed corrugated channels over *Re* ranging from 10,000 ≤ *Re* ≤ 30,000 were examined under constant heat flux while ϕ ranged from 0 to 0.06.

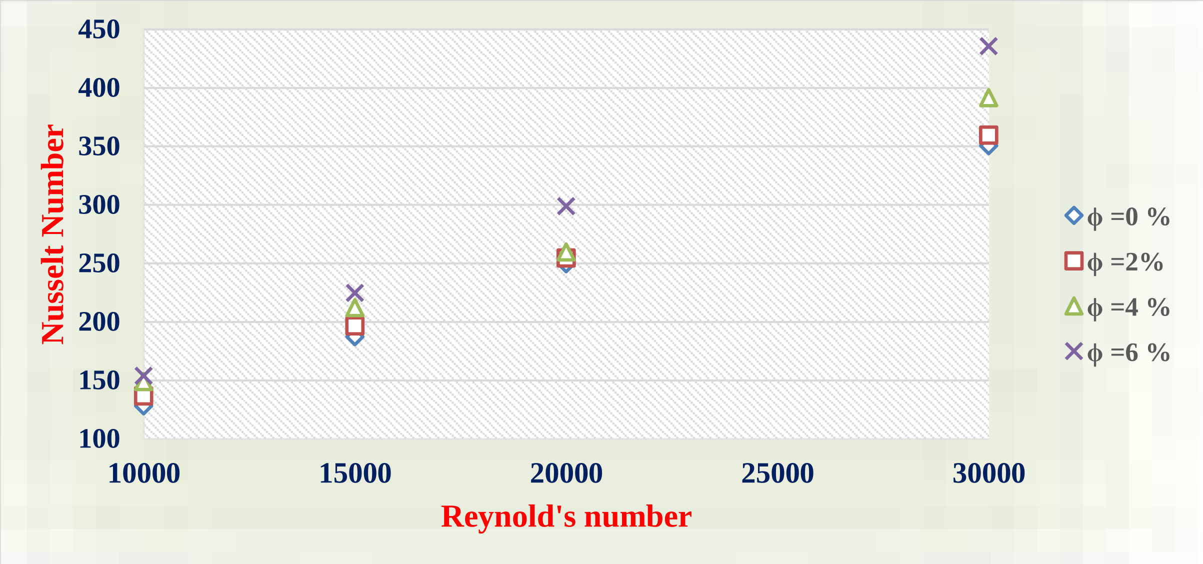

Figure 7 Value of Nusselt number calculated through CFD analysis at different Reynold's number for semi-circular + Triangular corrugated configuration under constant heat flux while ϕ ranged from 0 to 0.06

VI. Conclusions and Scope of future work

6.1. Conclusions

Corrugated passages are one of the common passive techniques used extensively in the manufacture of plate heat exchangers. Three parameters, including Reynolds number, nanofluid concentration, and channel configurations of the corrugated channel modules, were systematically varied in this analysis. The advantages obtained from these passages were evaluated by the Nusselt number. It is important to draw the following conclusions:

- As opposed to the traditional corrugated channel, a major heat transfer gain was associated with the proposed corrugated channel. Based on the nanofluid concentration, Reynold number and channel configuration of the corrugated channel, the average Nusselt number increased by a factor of 2.2 up to 2.8 compared to that of the semi-circular corrugated channel.
- Compared with the effect of Reynold's number variation, especially at high Reynolds number, nanoparticle concentration variation in the base fluid seemed to have a small effect on Nusselt number. As Ø increases, the value of the Nusselt number increases, which typically obtains higher values at higher nanoparticle concentrations.
- Compared with the effect of channel configuration variation, triangular corrugated channel was the best Nusselt numbers, followed by the shape (Semi-circular + Triangular), and Semi-circular corrugated channel with the lowest results.
- Reasonable consensus was seen in accordance with the available literature; evidence from the current investigation revealed a pattern close to that recorded in the literature, and variations could be attributed to differences in geometries of comparable channels and irregularities in measurements.

6.2. Scope of future work

- Researchers investigated over and over the process of passive thermal flow control. In order to fill the open space, active thermal flow control methods or passive and active thermal flow control methods of compounding should be further considered.
- More studies on pressure losses need to be carried out, especially when using the technique of compound thermal flow control in to extend the thermal efficiency tests further.
- The flow characteristics and heat transfer characteristics inside the corrugated channels have been investigated by researchers, but the requisite analytical correlations have not been established to quantify the heat transfer rate and pressure losses to determine thermal efficiency. That is to add, researchers can pay more effort to present correlations to lead heat exchanger designers so that heat exchanger designers or manufacturers can use them.

REFERENCES

[1] Abed Waleed Mohammed, Ahmed Mohammed Abed, 2008, "Numerical Study Of Laminar Forced Convection Heat Transfer And Fluid Flow Characteristics In A Corrugated Channel", Journal Of Engineering And Development, Volume 14, Issue 3, PP 70-85.

[2] Goodarzi Marjan, Amiri Ahmad, Goodarzi Mohammad Shahab, Safaei Mohammad Reza, Karimipour Arash, Languri Ehsan Mohseni, Dahari Mahidzal, 2008, "Investigation Of Heat Transfer And Pressure Drop Of A Counter Flow Corrugated Plate Heat Exchanger Using MWCNT Based Nanofluids", International Communication In Heat And Transfer, PP 172-179.

[3] Khan Mohd. Rehan, Kumar Ajeet, 2009 "An Experimental Study of Exergy in A Corrugated Plate Heat Exchanger'', International Journal of Mechanical Engineering and Technology, Volume 6, Issue 11, PP – 16-22.

[4] Rao B. Sreedhara, Svaron, Krishna MVS Murali, Sastry R C, 2009, "Experimental Studies on Pressure Drop In a Sinusoidal Plate Heat Exchanger: Effect of Corrugation Angle "International Journal of Research in Engineering and Technology, Volume 3, Issue 2, PP 121-126.

[5] Kumar Ashish, Rai Ajeet Kumar, Sachanvivek, 2010 "An Experimental Study of Heat Exchanger in A Corrugated Plate Heat Exchanger'', International Journal of Mechanical Engineering and Technology, Volume 5, Issue 9, PP.286-292.

[6] Rao B. Sreedhara, S Varun, G D Suryawanshi, Prof. Sastry. RC, 2010, "Experimental Heat Transfer Studies Of Water In Corrugated Plate Heat Exchanger And Effects Of Corrugation Angle'', International Journal Of Scientific Engineering And Technology, Volume 3, Issue 7, PP 902-905.

[7] Yin Jixiang, Yang Gang, Li Yang, 2011, "The Effect of Wavy Plate Phase Shift Flow and Heat Transfer Characteristics in Corrugated Channels", International Conference on Advances in Energy Engineering, PP-1566-1573.

[8] Faizal M., Ahmed M. R., 2012, "Experimental Studies On A Corrugated Plate Heat Exchanger For Small Temperature Difference Applications", Experimental Thermal And Fluid Science, PP-242-248

[9] Pandey S. D., Nemav. K., 2012 "Investigation Of The Performance Parameters Of An Experimental Plate Heat Exchanger In Single Phase Flow'', International Journal Of Energy Engineering, Volume 1, Issue 1, PP-19-24.

[10] Sapkal Dnyaneshwar B., Deshmukhsamir J, Kolhekarrucha R., 2013, "Computer Aided Design CFD Analysis Of Plate Heat Exchanger", International Journal Of Innovative And Emerging Research In Engineering, Volume 1, Issue 1, PP-88-92.

[11] Giurgiua Oana, Pleșaa Angela, Socaciualavinia, 2014, "Plate Heat Exchangers - Flow Analysis through Mini Channels", Energy and Environment, PP-244-251.

[12] Pandita Manoj Kumar, Prof Gupta Ashok Kumar, Lahadotiyakailash Kumar, 2014, "CFD Analysis Of Single Shell And Single Tube Heat Exchanger And Determining The Effect Of Baffle Angle On Heat Transfer", International Journal Of Engineering Science And Research Technology, Volume 4, Issue 6, PP-445-450.

[13] R Melvinraj C, C Vishal Varghese, Wilson Vicky, Thazhatha Thomas Jerry, Kakkasserymithunk, Simon J, 2014, "Comparative Study Of Heat Exchangers Using CFD, Int. Journal Of Engineering Research And Applications" Volume 4, Issue 5, PP-118-120.

[14] Kansal Santosh, Fatehmohd. Shabahat, 2015, " Design And Performance Evaluation Of Shell And Tube Heat Exchanger Using CFD Simulation", International Journal Of Research And Technology, Volume 3, Issue 7, PP-1663-1666.

[15] Hasanpour A., M. Farhadi, and K. Sedighi. "Experimental heat transfer and pressure drop study on typical, perforated, V-cut and U-cut twisted tapes in a helically corrugated heat exchanger." International Communications in Heat and Mass Transfer 2016; 70; 126-136.5

[16] Johnson G., Aikaterini A. Mouza, and Spiros V. Paras. "Flow and heat transfer prediction in a corrugated plate heat exchanger using a CFD code." Chemical Engineering & Technology: Industrial Chemistry-Plant Equipment-Process Engineering-Biotechnology 2017; 29(8): 923-930.

[17] R K Ajeeland W S I W Salim1, 2017, " A CFD study on turbulent forced convection flow of Al_2O_3- water nanofluid in semi-circular corrugated channel", IOP Conf. Series: Materials Science and Engineering 243 (2017) 012020.

[18] Junqi, Dong, Zhang Xianhui, and Wang Jianzhang. "Experimental Study on Thermal Hydraulic Performance of Plate-Type Heat Exchanger Applied in Engine Waste Heat Recovery." Arabian Journal for Science and Engineering 2018; 43(3): 1153-1163.

[19] Sharif Asal, Bernd Ameel, Ilya T'Jollyn, et al. "Comparative performance assessment of plate heat exchangers with triangular corrugation." Applied Thermal Engineering 2018.

[20] Khavin G. Simulation and Design of Welded Plate Heat Exchangers with Channels of Different Corrugation Height. In Design, Simulation, and Manufacturing: The Innovation Exchange 2018; 453-462.